ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

COMPARATIVE STUDY ON DESIGN AND ANALYSIS OF MULTISTOREYED BUILDING (G+10) BY STAAD.PRO AND ETABS SOFTWARE'S

D.Ramya*, A.V.S.Sai Kumar

**M.Tech, Dept.Of Civil Engg, Nalanda institute of technology,Sattenapalli ,A.P., India-515002.*
Asst.Professor, Dept. Of Civil Engg, , Nalanda institute of technology,Sattenapalli, A.P., India-515002

ABSTRACT

Structural Analysis is a branch which involves in the determination of behaviour of structures in order to predict the responses of real structures such as buildings, bridges, trusses etc. Under the improvement of expected loading & external environment during the service life of structure.

The results of analysis are used to verify the structure fitness for use. Computer software's are also being used for the calculation of forces, bending moment, stress, strain &deformation or deflection for a complex structural system.

The principle objective of this project is the comparative study on design and analysis of multi-storeyed building (G+10) by STAAD.Pro and ETABS software's

STAAD.Pro is one of the leading softwares for the design of structures. In this project we had analysed the G+10 building for finding the shear forces, bending moments, deflections & reinforcement details for the structural components of building (such as Beams, columns & slabs) to develop the economic design.

ETABS is also a leading design software in present days used by many structural designers. Here we had also analysed the same structure using **ETABS** software for the design. Finally we will made an attempt to define the economical section of G+10 multi-storeyed building using both **STAAD.Pro** and **ETABS** comparatively.

Key Words: *multi storied, seismic analysis, staad, etabs.*

INTRODUCTION

The full form of **STAAD** is STRUCTURAL AIDED ANALYSIS AND DESIGN. It was developed by Research Engineers International in Yorba Linda,CA later it was sold to Bentley systems in late 2005. STAAD.Pro is an analysis & design software package for structural engineering used in performing the analysis & design of wide variety of types of structures. It allows structural engineers to analyze& design virtually any type of structure through its flexible Modeling environment, advanced features & fluent data collaboration.

STAAD.Pro may be utilized for analyzing and designing practically all types of structures – buildings, bridges, towers, transportation, industrial and utility structures.

ETABS is the Acronym of EXTENDED 3D ANALYSIS OF BUILDING SYSTEMS, is software developed by Computers and Structures, Inc. (CSI); a Berkeley, California based engineering software company founded in 1975. ETABS is an engineering software product that can be used to analyze and design multi-story buildings using grid-like geometry, various methods of analysis and solution techniques, considering various load combinations.

ETABS can also handle the largest and most complex building models, including a wide range of nonlinear behaviors, making it the tool of choice for structural engineers in the building industry. ETABS can be effectively used in the analysis and design of building structures which might consists of structural members like beams, columns, slabs, shear walls etc, With ETABS you can easily apply various construction materials to your structural members like concrete, structural steel, Reinforced Concrete etc. ETABS automatically generates the self-weights and the resultant gravity and lateral loads.

LITERATURE REVIEW

Dharne Sidramappa Shivashaankar, Patil Raobahdur Yashwant presents the various limitations in design and construction practices along with the feedback to overcome the limitations and make the structures safer to take the earthquake forces. The paper focuses on software used in the civil engineering for analysis and design, construction methods/practices, use of materials, types of structures, experiments for earthquake studies, quality control parameters etc.

Prashanth.P, Anshuman.S, Pandey .R .K, Arpan Herbert present day leading design software's in the market. Many design company's use these software's for their project design purposes. So, this project mainly deals with the comparative analysis of the results obtained from the design of a regular and a plan irregular (as

per IS 1893) multi storey building structure when designed using STAAD.Pro and ETABS software's separately. These results will also be compared with manual calculations of a sample beam and column of the same structure designed as per IS 456.

Chaitanya Kumar J.D, Lute analysed G+11 storey residential building with precast reinforced concrete load bearing walls. The structural system consists of load bearing walls and one-way slabs for gravity and lateral loads have been taken for analysis using ETABS. Various wall forces, displacements and moments have been worked out for different load combinations. Data base is presented for the worst load combination. This work is limited to the analysis of structural elements only not the connection details.

Ismail Sab, Prof .S.M. Hashmi , generated a 3D analytical model of twelve storeyed buildings for different buildings Models and analysed using structural analysis tool ETABS. To study the effect of infill, ground soft, bare frame and models with ground soft having concrete core wall and shear walls and concrete bracings at different positions during earthquake; seismic analysis using both linear static, linear dynamic (response spectrum method) has been performed. The analytical model of the building includes all important components that influence the mass, strength, stiffness and deformability of the structure.

Swati D.Ambadkar, Vipul S. Bawner, analysed G +11 building by using STAAD PRO. Analysis is done for various variations such as 1) Terrain with few or no obstructions having heights below 1.5 m. 2) Terrain with obstructions having heights between 1.5 to 10 m. 3) Terrain with numerous closely spaced obstructions having the size of building structures up to 10 m in height.4) Terrain with numerous large high closely spaced obstructions. According to Internal Pressure Coefficients (*Cpi*) provided for that various variations. This analysis is done for wind speed 44 m/s, 47 m/s, 50 m/s. Results obtained from STAAD-PRO analysis are used for obtaining significant relations of moments, forces and displacement with wind speeds. Moments, forces and displacement obtained from all cases are compared with wind speeds, according to their percentage of opening provided for various variations.

BUILDING DATA FOR ANALYSIS

The proposed building considered for the project is of G + 7. The considered building data is furnished below:
Building information:
Building importance category : All other buildings = 1.0
Number of storeys : G +7, 8 Storey building.
Length of the building in X direction : 15.0m
Length of the building in Y direction : 15.0m
Inter storey height of the building : 3.5m (storey height to storey height)
Floor load : Dead load considered for the building is 6.5kPa.
Live load considered for the building is 5.0kpa.
Interior wall : dead load of 4.75kpa.
External wall : dead load of 2.40kpa.
Roof is at the level of 1.5m above with dead load of 7.71kpa and live load of 0.25kpa.
Structure in both X direction and in Y direction the resisting systems are considered and are then subjected to earthquake load and identification of safety conditions are made.
Structure in X direction : Reinforced concrete shear wall / reinforced masonry shear wall.
Structure in Y direction : Reinforced concrete shear wall / reinforced masonry shear wall.
The structure was analysed for all the cases of zones II, III, IV and V and for soil conditions of rocky and medium soils. It was not safe to use the masonry shear wall systems in soft soil conditions and hence the project was restricted for medium and rocky soils.
Wind load data considered for the analysis of building:
Wind region considered for the building is zone I.
Basic wind speed of the wind in zone I is of 33.0m/s.
Terrain category of the building is built up towns.
Site shape is flat in conditions and not hill slopes.
The analysis is of the structure was made using resist software used for the analysis of multi storied building in Indian conditions for application of earthquake intensities for all zones and also for application of wind effects also.
The results obtained for the analysis of the building are given in the next section.

ANALYSIS AND DESIGN OF BUILDING

ANALYSIS AND DESIGN OF G + 10 BUILDING USING STAAD. Pro

Step - 1 : Creation of nodal points.
Based on the column positioning of plan we entered the node points into the STAAD file

Step - 2 : Representation of beams and columns.
By using add beam command we had drawn the beams and columns between the corresponding node points.

Step - 3: 3D view of structure.
Here we have used the Transitional repeat command in Y direction to get the 3D view of structure.

Step - 4: Supports and property assigning.
After the creation of structure the supports at the base of structure are specified as fixed. Also the materials were specified and cross section of beams and columns members was assigned.

Step - 5: 3D rendering view.
After assigning the property the 3d rendering view of the structure can be shown

Step - 6: Assigning of seismic loads.
In order to assign Seismic loads firstly we have defined the seismic loads according to the code **IS 1893:2002** with proper floor weights. Loads are added in load case details in +X,-X, +Z,-Z directions with specified seismic factor.

Step - 7: Assigning of wind loads.
Wind loads are defined as per **IS 875 PART 3** based on intensity calculated and exposure factor. Then loads are added in load case details in +X,-X, +Z,-Z directions.

Step - 8: Assigning of dead loads.
Dead loads are calculated as per **IS 875 PART 1** for external walls, internal walls, parapet wall including self-weight of structure.

Step - 9: Assigning of live loads.
Live loads are assigned for every floor as 4 kN/m^2 based on **IS 875 PART 2**.

Step - 10: Adding of load combinations.
After assigning all the loads, the load combinations are given with suitable factor of safety as per **IS 875 PART 5.**

Step - 11: Analysis.
After the completion of all the above steps we have performed the analysis and checked for errors.

Step - 12: Design.
Finally concrete design is performed as per **IS 456: 2000** by defining suitable design commands for different structural components. After the assigning of commands again we performed analysis for any errors.

ANALYSIS AND DESIGN OF G + 10 BUILDING USING ETABS

Step - 1 : Step by Step procedure for ETABS Analysis
The procedure carried out for Modelling and analyzing the structure involves the following flow chart.

Step - 2 : Creation of Grid points & Generation of structure
After getting opened with ETABS we select a new model and a window appears where we had entered the grid dimensions and story dimensions of our building. Here itself we had generated our 3D structure by specifying the building details in the following window.

Step - 3: Defining of property
Here we had first defined the material property by selecting define menu → material properties. We add new material for our structural components (beams, columns, slabs) by giving the specified details in defining. After that we define section size by selecting frame sections as shown below & added the required section for beams, columns etc.

Step - 4: Assigning of Property
After defining the property we draw the structural components using command menu → Draw line for beam for beams and create columns in region for columns by which property assigning is completed for beams and columns.

Step - 5: Assigning of Supports
By keeping the selection at the base of the structure and selecting all the columns we assigned supports by going to assign menu → joint\frame → Restraints (supports) →fixed.

Step - 6: Defining of loads

In STAAD program we define only seismic and wind loads where as in ETABS all the load considerations are first defined and then assigned. The loads in ETABS are defined as using static load cases command in define menu

Step - 7: Assigning of Dead loads

After defining all the loads dead loads are assigned for external walls (D230) , internal walls

Step - 8: Assigning of Live loads

Live loads are assigned for the entire structure including floor finishing.

Step - 9: Assigning of wind loads

Wind loads are defined and assigned as per IS 875 1987 PART 3 by giving wind speed and wind angle in X,X1,Z & Z1 directions as 0 , 180 , 90 , 270 respectively

Step - 10: Assigning of Seismic loads

Seismic loads are defined and assigned as per IS 1893: 2002 by giving zone, soil type, and response reduction factor in X and Y directions

Step - 11: Assigning of load combinations

Load combinations are given as mentioned in STAAD. Pro based on IS 875 1987 PART 5 using load combinations command in define menu

Step - 12: Analysis

After the completion of all the above steps we have performed the analysis and checked for errors.

Step - 13: Design

After the completion of analysis we had performed concrete design on the structure as per IS 456:2000. For this go to Design menu →concrete design →select design combo. After this again go to design menu →concrete frame design →start design \ check of structure then ETABS performs the design for every structural element.

RESULTS

COMPARISON OF SHEAR FORCE AND BENDING MOMENT VALUES OF A SAMPLE COLUMN:

Loading	Forces	STAAD .Pro	ETABS
DL	Fx	885.226	885.12
	Fy	3.499	3.45
	Fz	4.770	4.75
	Mx	0.112	0.109
	My	8.849	8.84
	Mz	1.903	1.902
LL	Fx	333.388	333.367
	Fy	2.735	2.732
	Fz	2.720	2.72
	Mx	0.067	0.066
	My	5.133	5.132
	Mz	2.308	2.307
EQ(along length)	Fx	16.102	16.101
	Fy	1.081	1.08
	Fz	0.778	0.77
	Mx	0.320	0.32
	My	0.198	0.197
	Mz	4.777	4.76
EQ(along width)	Fx	32.737	32.736
	Fy	0.018	0.018
	Fz	4.047	4.047
	Mx	0.061	0.06
	My	0.766	0.766
	Mz	0.026	0.025
WL(along length)	Fx	131.961	131.96
	Fy	20.679	20.67
	Fz	0.644	0.644
	Mx	2.409	2.408
	My	1.77	1.777
	Mz	83.994	83.99
	Fx	92.540	92.53

WL(along width)	Fy	0.278	0.276
	Fz	17.365	173.35
	Mx	1.717	1.716
	My	4.360	4.359
	Mz	0.085	0.084

COMPARISON OF SHEAR FORCE AND BENDING MOMENT VALUES OF A SAMPLE BEAM

Loading	Forces	STAAD .Pro	ETABS
DL	Fx	2.228	2.227
	Fy	25.024	25.023
	Fz	0.095	0.095
	Mx	0.039	0.038
	My	0.236	0.235
	Mz	20.282	20.28
LL	Fx	1.321	1.320
	Fy	11.802	11.801
	Fz	0.051	0.051
	Mx	0.048	0.047
	My	0.132	0.131
	Mz	11.767	11.76
EQ(along length)	Fx	0.194	0.194
	Fy	0.502	0.502
	Fz	0.021	0.021
	Mx	0.078	0.077
	My	0.054	0.053
	Mz	1.164	1.163
EQ(along width)	Fx	1.09	1.08
	Fy	2.709	2.708
	Fz	0.011	0.011
	Mx	0.001	0
	My	0.033	0.032
	Mz	6.281	6.280
WL(along length)	Fx	0.636	0.636
	Fy	0.411	0.411
	Fz	0.587	0.586
	Mx	0.856	0.84
	My	1.118	1.117
	Mz	1.013	1.012
WL(along width)	Fx	2.566	2.565
	Fy	11.01	11.01
	Fz	0.34	0.34
	Mx	0.058	0.057
	My	0.631	0.632
	Mz	25.462	25.462

COMPARISON

S.No	Point of Comparison	Software		Remarks
		STAAD.Pro	ETABS	
1.	Time	It takes less time	It takes slightly more time.	STAAD is very easy to learn& work.
2.	Accuracy	Less accurate.	More accurate	STAAD is accurate for both analysis and design
3.	Flexibility	User friendly	Learners choice	***
4.	Present day status	Most of the designers are using this software	Not preferred like STAAD	STAAD is more preferred because of its flexibility and ease of workability
5.	Steel	122.58 tons	111.24 tons	***
6.	Concrete	1086 cum	1086 cum	***

CONCLUSIONS

- STAAD.Pro software is more flexible to work compared to the ETABS software.
- The quantity of steel requirement is *9.25%* less for the design of G+10 multi-storied building using ETABS compared with the STAAD analysis.
- The quantity of concrete requirement is same for the design of the G+10 multi-storied building using both STAAD and ETABS analysis.
- By the intensive study of "Comparative study on Analysis and Design of G+10 multi-storied building by both STAAD and ETABS software's" the *"economical sections"* was developed by ETABS software.

REFERENCES

1. Design And Practical Limitations In Earthquake Resistant Structures And Feedback . INTERNATIONAL JOURNAL OF CIVIL ENGINEERING AND CIVIL ENGINEERING (IJCIET), Volume 5, Issue 6, June (2014), pp. 89-93
2. Comparison of design results of a Structure designed using STAAD and ETABS Software, INTERNATIONAL JOURNAL OF CIVIL AND STRUCTURAL ENGINEERING, Volume 2, No 3, 2012
3. Analysis of multi storey building with precast load bearing walls. INTERNATIONAL JOURNAL OF CIVIL AND STRUCTURAL ENGINEERING, Volume 4, No 2, 2013
4. Analysis and Design of shopping mall against lateral forces.INTERNATIONAL JOURNAL OF ENGINEERING SCIENCE INVENTION,Volume 3 Issue 4 April 2014.
5. Lateral Load Analysis of Shear Wall and Concrete Braced Multi-Storeyed R.C Frame with the Effect of Ground Soft Storey. IJSRD - International Journal for Scientific Research & Development, Vol. 2, Issue 09, 2014.
6. Behavior of multistoried building under the effect of wind load. INTERNATIONAL JOURNAL OF APPLIED SCIENCES AND ENGINEERING RESEARCH, Vol. 1, Issue 4, 2012.